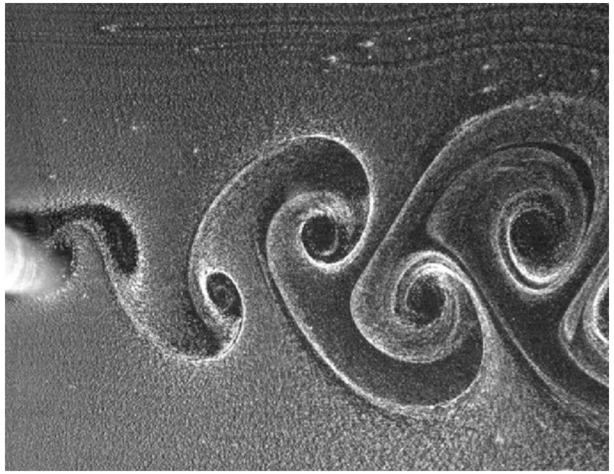
清華大學深圳国际研究生院
Tsinghua Shenzhen International Graduate School

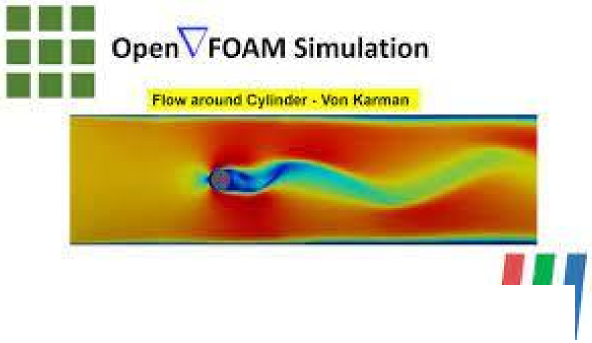
基于Precice平台openFOAM和Calculix的流固耦合分析

CAE汇报　　汇报人：吴华晓　　时间：2023年06月09

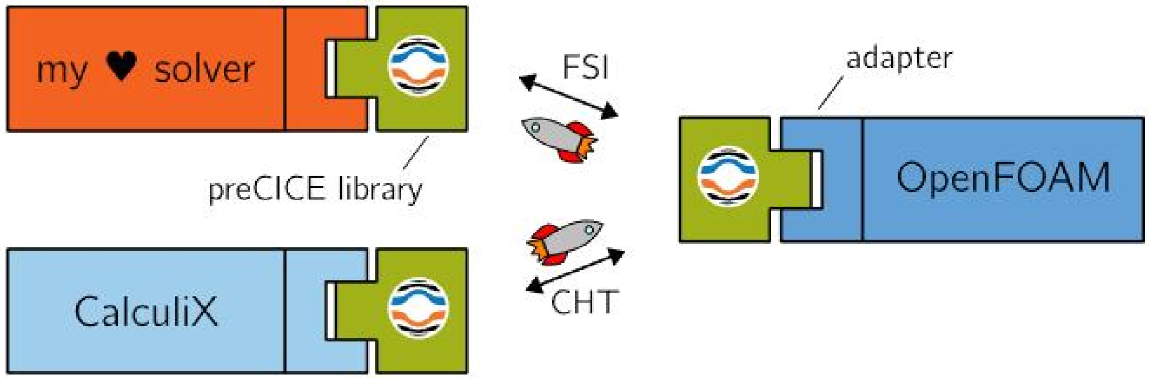
Outline for presentation

1. 研究背景
2. 三种开源工具
3. 案例介绍
4. 仿真数据可视化
5. 结论和后续





preCICE



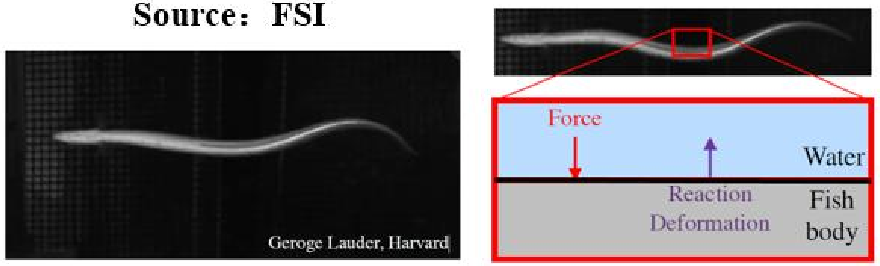
CalculiX

1、研究背景

什么是流固耦合?

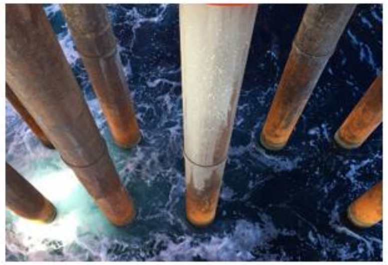
- 流固耦合是流体和固体间的**交互作用**
- ➢ 流体产生**压强**引起固体**变形**
- ➢ 固体**形变**后改变流场的**边界条件**

Source：FSI



自然环境

土木工程



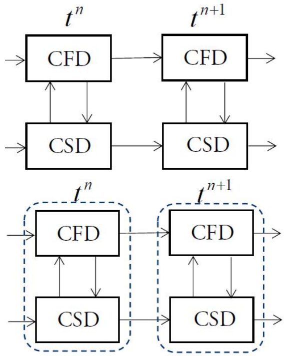
海洋工程

1、研究背景

如何研究流固耦合?

基本概念

- 流固耦合分为**单向耦合**和**双向耦合**
- 单向忽略了固体变形对流场的影响
- 双向由分为**弱耦合**和**强耦合**形式
- 弱耦合在一个时间步只交换**一次**信息
- 强耦合在一个时间步**多次交换信息**直至收敛
- 算法分为统一法和分区耦合算法



数值工具

Academic Codes [edit]

- Stochastic Immersed Boundary Methods in 3D, P. Atzberger, UCSB
- Immersed Boundary Method for Adaptive Meshes in 3D, B. Griffith, NYU.
- Immersed Boundary Method for Uniform Meshes in 2D, A. Fogelson, Utah
- IFLS, IFL, TU Braunschweig

Commercial Codes [edit]

- Abaqus Multiphysics Coupling
- AcuSolve FSI applications
- ADINA FSI homepage Archived 2021-04-28 at the Wayback Machine
- Ansys' FSI homepage
- Altair RADIOSS
- Autodesk Simulation CFD
- Simcenter STAR-CCM+ from Siemens Digital Industries Software
- CoLyX - FSI and mesh-morphing from EVEN - Evolutionary Engineering AG
- Fluidyn-MP FSI Multiphysics Coupling Archived 2014-05-17 at the Wayback Machine
- COMSOL FSI homepage
- MpCCI homepage
- MSC Software MD Nastran
- MSC Software Dytran
- FINE/Oofelie FSI: Fully integrated and strongly coupled for better convergence Archived 2012-04-01 at the Wayback Machine
- LS-DYNA Home Page
- Fluidyn-MP FSI: Fluid-Structure Interaction
- CompassFEM Tdyn
- CompassFEM SeaFEM
- Cradle SC/Tetra CFD Software
- PARACHUTES FSI HomePage

OpenFOAM

Calculix

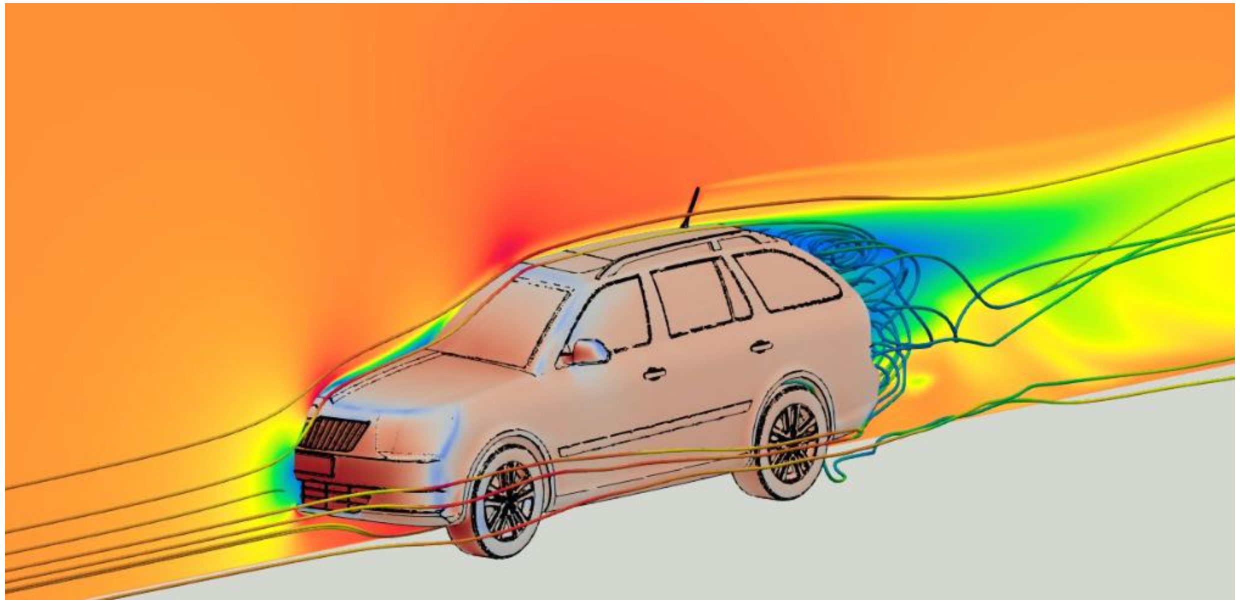
Open source codes [edit]

- solids4Foam, a toolbox for OpenFOAM with capabilities for solid mechanics and fluid solid interactions
- oomph-lib
- Elmer FSI page
- CBC.solve Biomedical Solvers
- preCICE Coupling Library
- SPHinXsys multi-physics library It provides C++ APIs for physical accurate simulation and aims to model coupled industrial dynamic systems including fluid, solid, multi-body dynamics and beyond with SPH (smoothed particle hydrodynamics), a meshless computational method using particle discretization.

2、三种开源工具

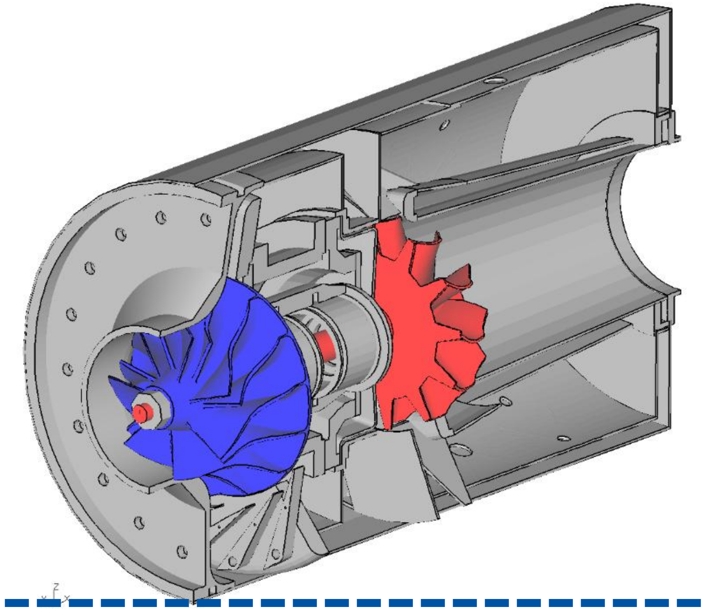
Fluid：OpenFOAM

- C++工具箱主要基于有限体积法用于开发定制的数值求解器，并提供预处理和后处理工具，用于解决连续介质力学问题，包括计算流体力学（CFD）也能解决多相流、粒子追踪、分子动力学等问题。



Solid：CalculiX

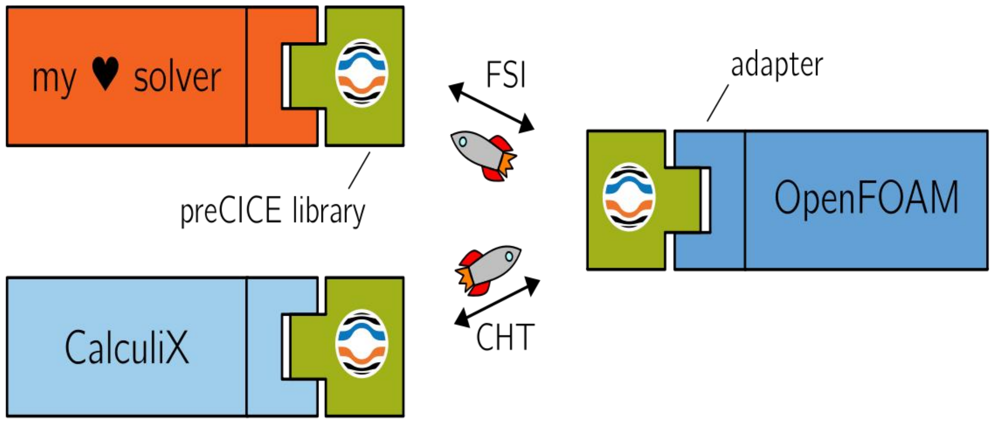
- CalculiX 是用来处理场问题的软件包
- 基于有限元法开发的求解器
- 能处理线性和非线性计算、能求解静力、动力和热力学问题
- 求解器使用的是Abaqus文件的输入格式

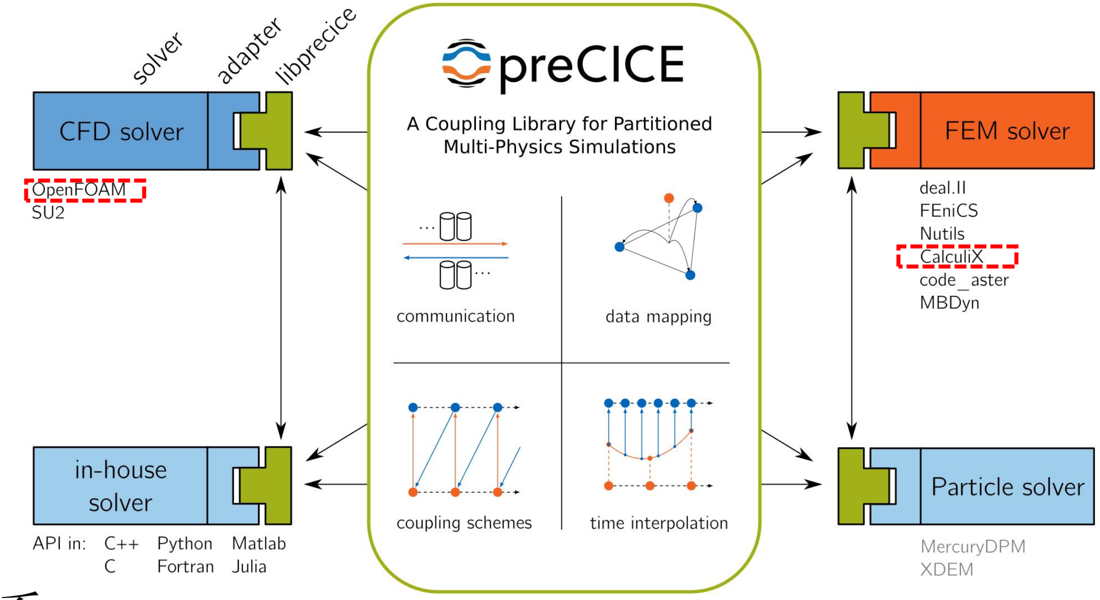


A Complete Example of FSI

Coupling：PreCICE

- preCICE 是一个专门用于**分区耦合算法**的开源耦合库。它能处理**流固耦合**和热传导问题等。它能耦合现有的物理仿真求解器，具有高度的灵活性，在复杂的多物理场耦合计算中保持良好的计算效率。





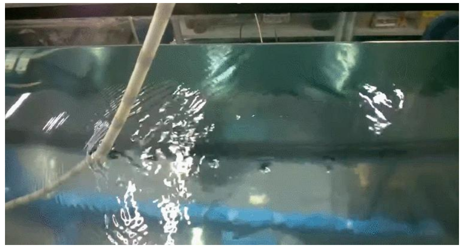
- communication:**界面信息**进行交互
- data mapping：规定两个计算域**空间网格点**的映射关系
- time interpolation: 规定不同场时间步长不协同时的**插分关系**
- coupling schems：规定信息交换的方式、如时间强耦合和弱耦合、空间上交换方式

3、案例介绍 实际问题

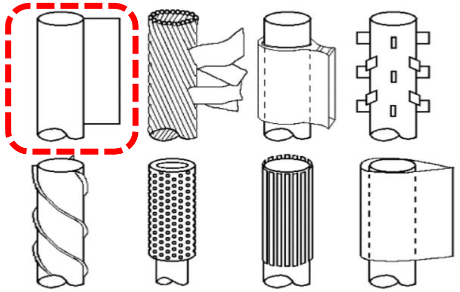
圆柱绕流涡脱会增加圆柱升阻力和响应幅值、需要设置绕流板来减阻。

实际FSI问题

圆柱绕流：



绕流板：



概念模型

物理模型：

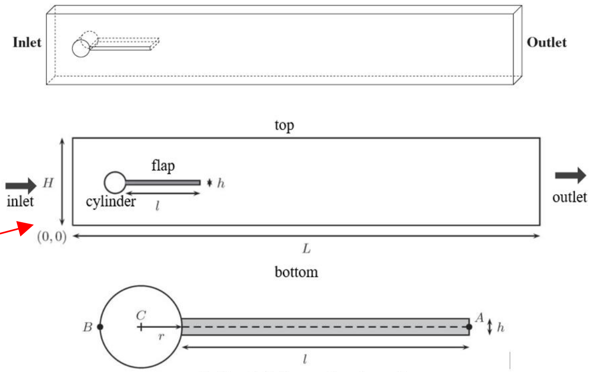


Fig. 4 计算域和结构物部分的几何尺寸[2]

几何信息

参数	L	H	C	r	l	h
数值(m)	2.5	0.41	(0.2,0.2)	0.05	0.35	0.02

物性信息

参数	$\rho^f\left(kg/m^3\right)$	$\nu\left(m^2/s\right)$	$\rho^s\left(kg/m^3\right)$	$E^s\left(kg/ms^2\right)$	ν^s
数值	1	1e-5	1e3	5.6e6	0.4

数学模型

流体：

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=\nabla\cdot\vec{V}=0$$

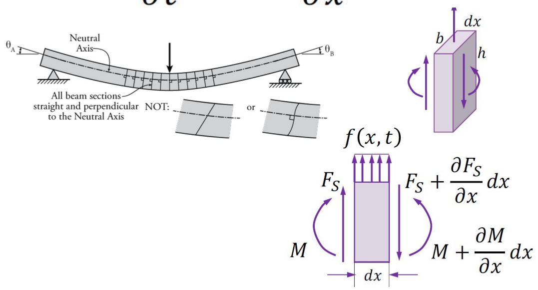
$$\rho\frac{Du}{Dt}=-\frac{\partial p}{\partial x}+\mu\vec{\nabla}^2u$$

$$\rho\frac{Dv}{Dt}=-\frac{\partial p}{\partial y}+\mu\vec{\nabla}^2v$$

$$\rho\frac{Dw}{Dt}=-\frac{\partial p}{\partial z}+\mu\vec{\nabla}^2w$$

固体：

$$\rho_s bh\frac{\partial^2 w}{\partial t^2}+EI\frac{\partial^4 w}{\partial x^4}=f(x,t)$$



离散数值模拟

流体域

基于有限体积法进行离散在openFOAM的PimpleFoam求解器中求解

交界面

计算信息在Precice平台进行交互

固体域

基于有限元进行离散在Calculix中进求解

3、案例介绍 网格优化

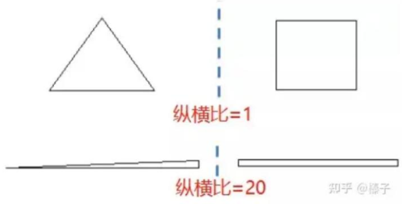
流体计算域分区和网格划分并没有**唯一选择**，但网格划分会影响到算法的收敛性和计算效率，因此需要对网格进行优化，网格质量标准见下表

网格质量评价标准

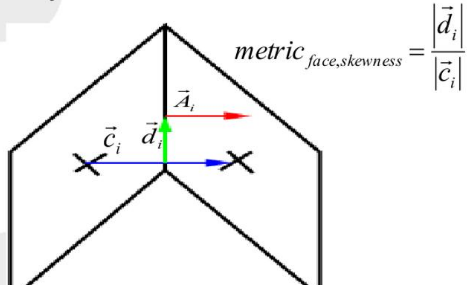
Aspect Ratio(纵横比)	其值越接近 1，质量越好
Non-orthogonality(非正交性)	其值越接近 0，说明网格质量越好
Skewness (偏斜系数)	其值越接近 0，说明网格质量约好
Parallel Deviation(对边偏差角)	其值越接近 0，说明网格质量越好。
Maximum Corner Angle (单元最大内角)	四边形，越接近 90 度越好
Jacobian Ratio (雅克比比率)	其值越接近 1，说明网格质量越好
Wraping Factor (翘曲系数)	其值越接近 0，说明网格质量越好
Element Quality (网格质量系数)	其值越接近 1，说明网格质量越好

Aspect Ratio纵横比（长宽比）

即对单元的三角形或四边形顶点计算长宽比（最长边/最短边）。数值≥1，等于1时表示质量最好，数值越大网格质量越差。结构分析中，纵横比应＜20，大于20将发生警告，大于1e6将发生错误，如图3所示。

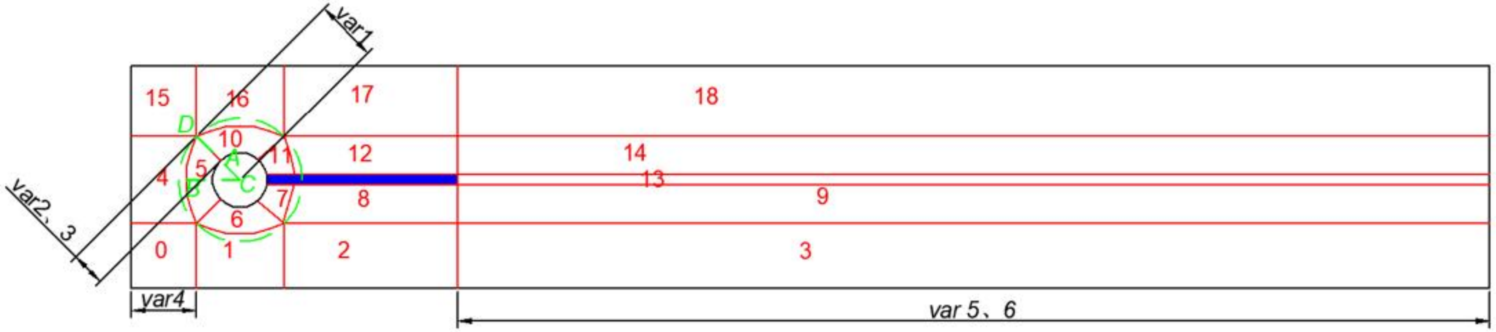


Face skewness is calculated as the distance from the face centre to the cell-centre to cell-centre face intersection point normalised by the distance from centroid of the cell to centroid of the adjacent cell



参数说明以及上下限

参数	参数说明	上下限
var1	圆柱外径 CD 与半径 r 比值	[1.5 3]
var2	圆柱周边 *AD* 段网格加密梯度	[1 5]
var3	圆柱外径 AD 段网格数	[5,15]
var4	Block0 横向网格数	[10,20]
var5	出口方向网格数	[30 100]
var6	出口方向网格加密梯度	[1 10]



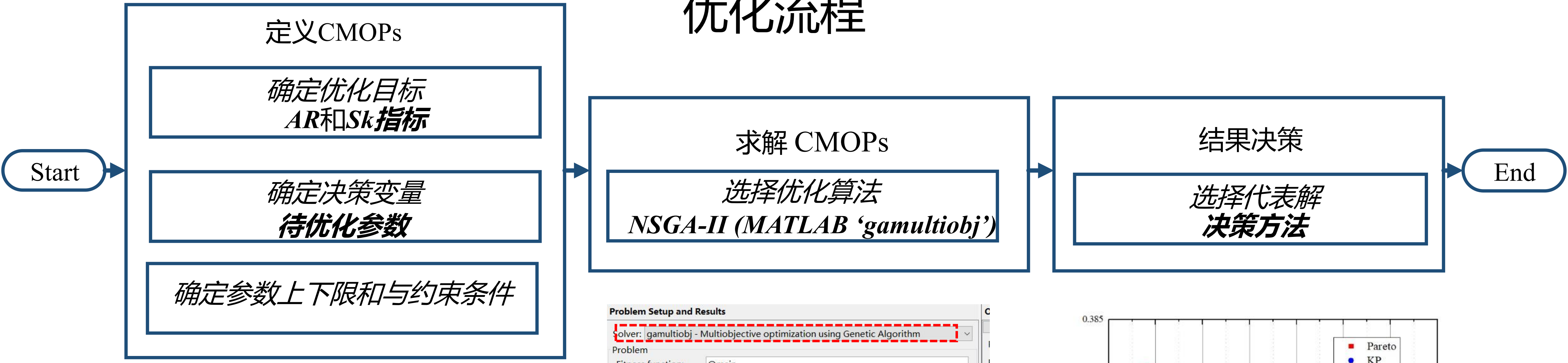
待优化参数说明

基于Matlab平台的NASA-II算法，以***Ar*和*Sk***为目标函数进行带约束的多目标优化问题(CMOPs)

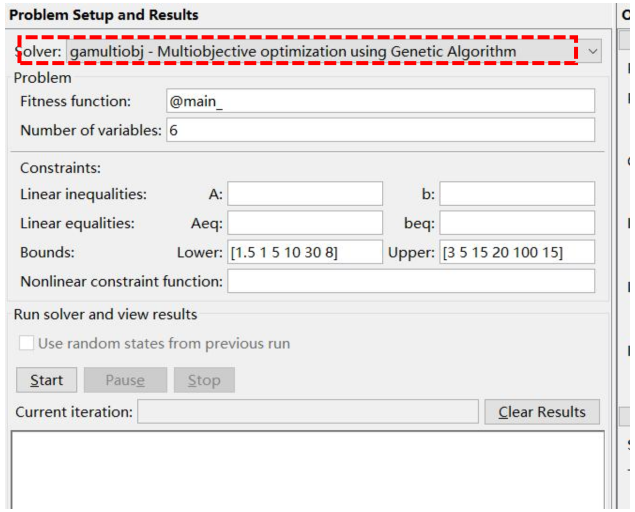
清华大学深圳国际研究生院 Tsinghua Shenzhen International Graduate School
海洋工程研究院 Institute for Ocean Engineering

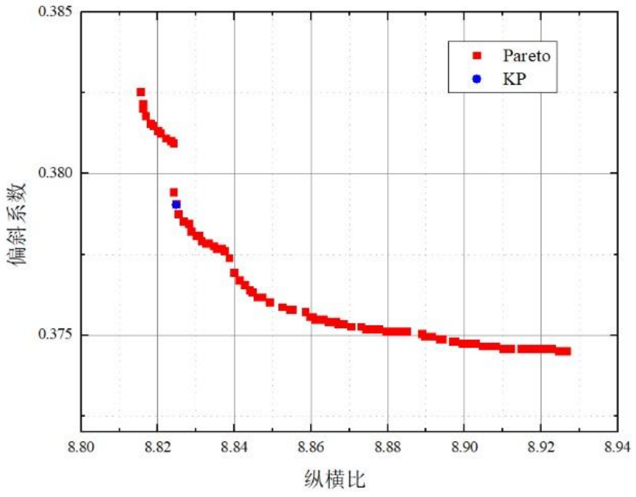
基于Matlab平台的NASA-II算法，以***Ar*和*Sk***为目标函数进行带约束的多目标优化

优化流程



$$
\begin{cases}
\text{minimize} \quad f_1(\mathrm{var}_1,\mathrm{var}_2,\mathrm{var}_3,\mathrm{var}_4,\mathrm{var}_5,\mathrm{var}_6) = Ar_{\max} \\
\qquad\qquad\quad f_2(\mathrm{var}_1,\mathrm{var}_2,\mathrm{var}_3,\mathrm{var}_4,\mathrm{var}_5,\mathrm{var}_6) = Sk_{\max} \\
s.t. \\
\qquad \mathrm{var}_1 \in [1.5,3], \mathrm{var}_2 \in [1,5], \\
\qquad \mathrm{var}_3 \in [5,15], \mathrm{var}_4 \in [10,20], \\
\qquad \mathrm{var}_5 \in [30,100], \mathrm{var}_6 \in [8,15]
\end{cases}
$$





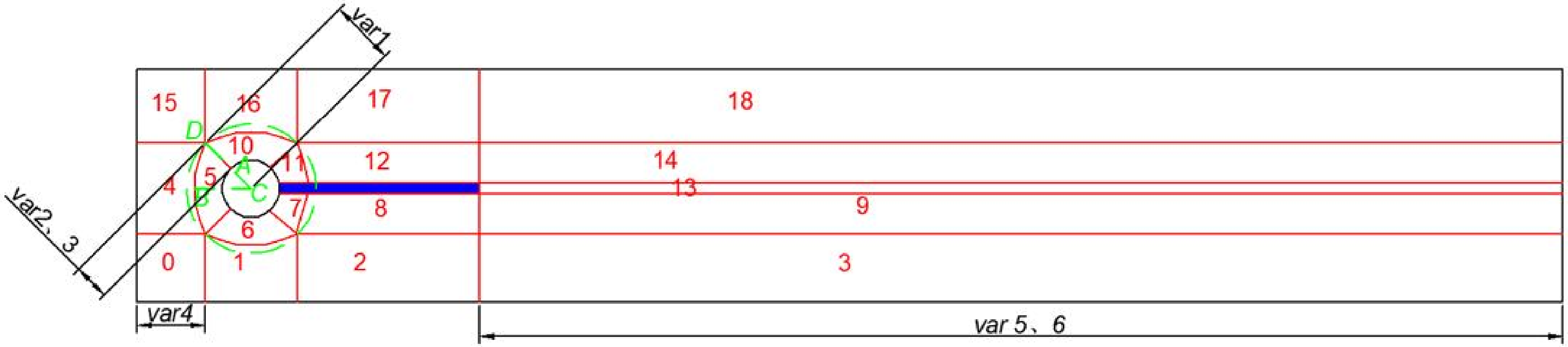
3、案例介绍

网格优化

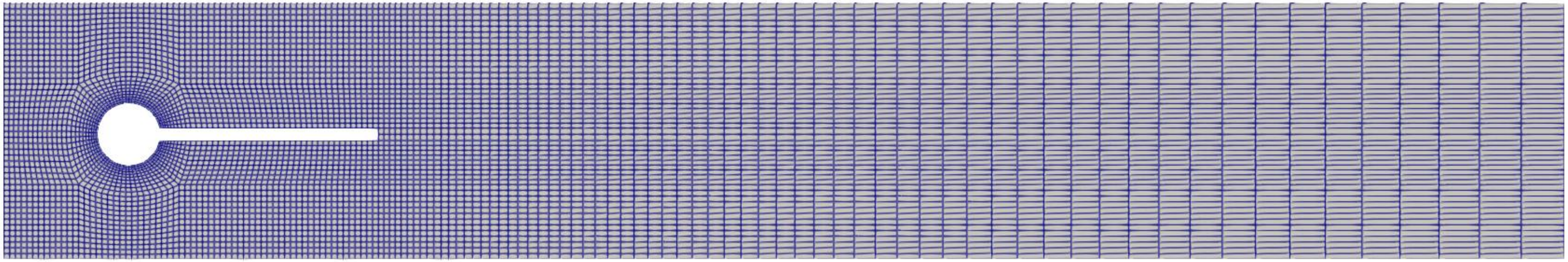
基于Matlab平台的NASA-II算法，以***Ar*和*Eq***为目标函数进行带约束的多目标优化。最优kp点对应的参数是[2.28 2.49 10 13 65 7.89]，将得到以下的网格。

参数说明以及上下限

参数	参数说明	上下限
var1	圆柱外径 CD 与半径 r 比值	[1.5 3]
var2	圆柱周边 *AD* 段网格加密梯度	[1 5]
var3	圆柱外径 AD 段网格数	[5,15]
var4	Block0 横向网格数	[10,20]
var5	出口方向网格数	[30 100]
var6	出口方向网格加密梯度	[1 10]



待优化参数说明



3、案例介绍

算例设置

算例设置分为三大块，流体域、固体域和交界面

本案例设置Re=5e4、湍流模型选用RANS，k-epsilon模型，湍流模型数学表达式详细见下式、在OpenFOAM中依次修改相应文件

k-ε model requires two additional partial differential equations to be solved

$$\mu_t = \rho c_\mu \frac{k^2}{\varepsilon}$$

Has been the most popular 2-equation model in the past

Turbulent Kinetic Energy

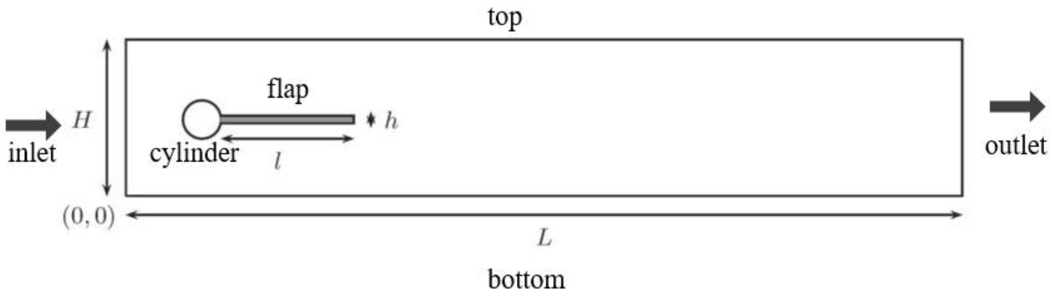
$$\underbrace{\rho U_i \frac{\partial k}{\partial x_i}}_{\text{Convection}} = \underbrace{\mu_t \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) \frac{\partial U_j}{\partial x_i}}_{\text{Generation}} + \underbrace{\frac{\partial}{\partial x_i} \left\{ (\mu_t / \sigma_k) \frac{\partial k}{\partial x_i} \right\}}_{\text{Diffusion}} - \underbrace{\rho \varepsilon}_{\text{Destruction}}$$

Dissipation Rate

$$\underbrace{\rho U_i \frac{\partial \varepsilon}{\partial x_i}}_{\text{Convection}} = \underbrace{C_{1\varepsilon} \left(\frac{\varepsilon}{k} \right) \mu_t \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) \frac{\partial U_j}{\partial x_i}}_{\text{Generation}} + \underbrace{\frac{\partial}{\partial x_i} \left\{ (\mu_t / \sigma_\varepsilon) \frac{\partial \varepsilon}{\partial x_i} \right\}}_{\text{Diffusion}} - \underbrace{C_{2\varepsilon} \rho \left(\frac{\varepsilon^2}{k} \right)}_{\text{Destruction}}$$

*(equations written for steady, incompressible flow w/o body forces)

$\sigma_k, \sigma_\varepsilon, C_{1\varepsilon}, C_{2\varepsilon}, C_\mu$ are empirical constants – tuned to canonical flows



计算域

几何信息

参数	L	H	C	r	l	h
数值(m)	2.5	0.41	(0.2,0.2)	0.05	0.35	0.02

流体求解器

初始条件

- epsilon
- k
- nut
- p
- pointDisplacement
- U
- vorticity

```
inlet
{
    // Time-varying inlet velocity
    type            uniformFixedValue;
    uniformValue    table
    (
        ( 0 ( 0 0 0 ) )
        ( 2. ( 5 0 0 ) )
        ( 100 ( 5 0 0 ) )
    );
}
```

- polyMesh
- dynamicMeshDict
- transportProperties
- turbulenceProperties

```
transportModel Newtonian;

nu nu [0 2 -1 0 0 0 0 ] 0.00001;
```

➢ 空气

```
simulationType RAS;

RAS
{
    RASModel        kEpsilon;

    turbulence      on;

    printCoeffs     on;
}

// ****************************
```

➢ 初始条件

时变速度

边界条件

U文件设置

```
flap
{
    type            movingWallVelocity;
    value           uniform (0 0 0);
}

frontAndback
{
    type            empty;
}

outlet
{
    type            zeroGradient;
}
```

P文件设置

```
outlet
{
    type            fixedValue;
    value           uniform 0;
}

frontAndback
{
    type            empty;
}

frontAndback
{
    type            empty;
}
```

- 其余壁面为noslip

离散格式

➢ 有限体积法进行离散

➢ 局部加速度项

➢ 对流项采用高斯线性迎风梯度

➢ 粘滞力项高斯线性修正

➢ 压力梯度项高斯线性

```
ddtSchemes
{
    default backward;
}
```

```
divSchemes
{
    default         none;
    div(phi,U)      bounded Gauss linearUpwind grad(U);
    div(phi,k)      bounded Gauss limitedLinear 1;
    div(phi,epsilon) bounded Gauss limitedLinear 1;
    div(phi,omega)  bounded Gauss limitedLinear 1;
    div(phi,v2)     bounded Gauss limitedLinear 1;
    div((nuEff*dev2(T(grad(U))))) Gauss linear;
    div(nonlinearStress) Gauss linear;
}

laplacianSchemes
{
    default         Gauss linear corrected;
}

gradSchemes
{
    default         Gauss linear;
}
```

迭代控制

➢ SIMPLE算法

➢ 主要是添加湍动能和耗散方向相关的参数设置

```
SIMPLE
{
    nNonOrthogonalCorrectors 1;//// 非正交校正次数
    consistent true; // SIMPLEC

    residualControl//// 残差控制
    {
        "(p|U|k|epsilon)" 1e-08;//残差控制方程和对应的阈值
    }
}

relaxationFactors
{
    p               0.7;// 压力方程的松弛因子
    U               0.7;// 速度方程的松弛因子
    k               0.7;//湍动能方程的松弛因子
    epsilon         0.7;// 粘性耗散率方程的松弛因子
}
```

3、案例介绍 算例设置

固体求解器

- add_material.inp
- all.msh
- fix1.nam
- flap.inp
- post_flap.fbd
- pre_flap.fbd
- surface.nam

```
*MATERIAL,NAME=User
*ELASTIC
5.6000E+06,0.4

*DENSITY
1.0000E+03
*EXPANSION
0.0000E+00

*SOLID SECTION,MATERIAL=User,ELSET=Eall
```

```
*BOUNDARY
Nfix1, 1, 6
** fix on all nodes for translation in z-dir, rotation about x- and y-axis
*BOUNDARY
Nall, 3, 3, 0
Nall, 4, 4, 0
Nall, 5, 5, 0
*CLOAD
 Nsurface, 1, 0.0
 Nsurface, 2, 0.0
 Nsurface, 3, 0.0
```

```
valu c 0.35
valu h 0.02
valu dz 1.
valu nd 0.005

pnt p1 0.25 0.19 -0.005
pnt p2 0.25 0.19 0.005
line l1 p1 p2 1
seta fix1 all
swep fix1 fix1 tra 0 h 0 1
seta fix1 all
comp fix1 do
```

- <u>有限元方法</u>进行离散
- 定义材料参数
- 定义边界条件
- 定义结构几何信息和节点拓扑结构
- 等等……

PreCICE

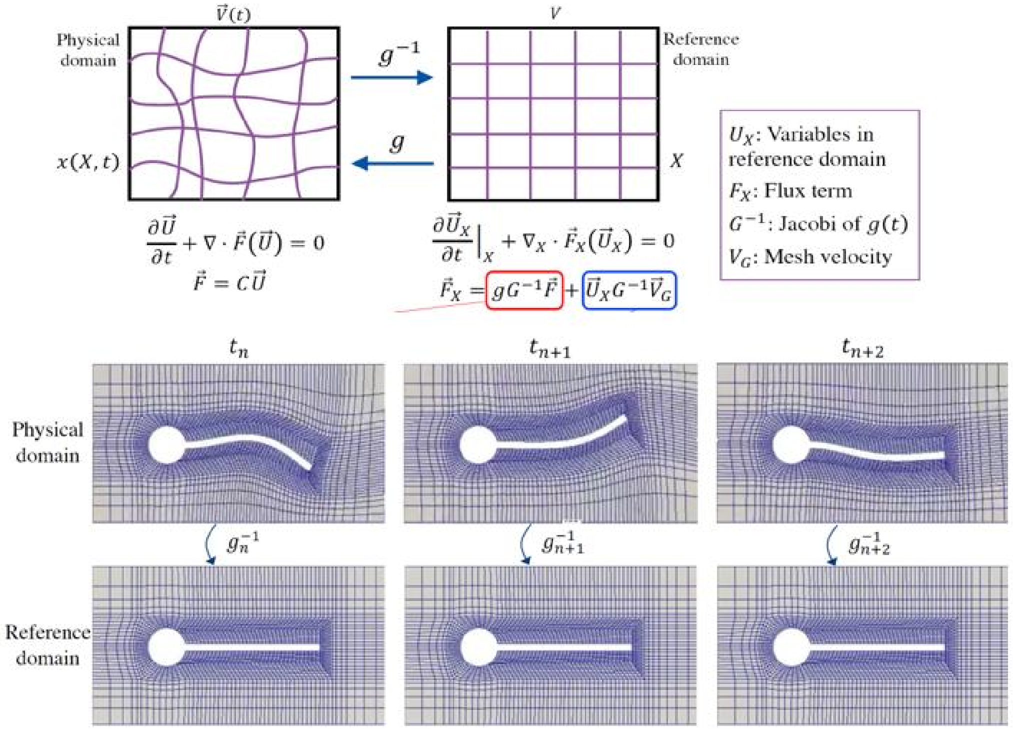
```
<participant name="Fluid">
   <use-mesh name="Fluid-Mesh-Faces" provide="yes"/>
   <use-mesh name="Fluid-Mesh-Nodes" provide="yes"/>
   <use-mesh name="Calculix_Mesh" from="Calculix"/>
   <write-data name="Forces0" mesh="Fluid-Mesh-Faces"/>
   <read-data name="Displacements0" mesh="Fluid-Mesh-Nodes"/>
   <mapping:rbf-thin-plate-splines direction="write" from="Fluid-Mesh-Faces" to="Calculix_Mesh" constraint="conservative" z-dead="true" />
   <mapping:rbf-thin-plate-splines direction="read" from="Calculix_Mesh" to="Fluid-Mesh-Nodes" constraint="consistent" />
</participant>

<participant name="Calculix">
   <use-mesh name="Calculix_Mesh" provide="yes"/>
   <read-data name="Forces0" mesh="Calculix_Mesh"/>
   <write-data name="Displacements0" mesh="Calculix_Mesh"/>
   <watch-point mesh="Calculix_Mesh" name="point1" coordinate="0.6;0.2;0." />
</participant>

<m2n:sockets from="Fluid" to="Calculix"/>

<coupling-scheme:serial-implicit>
     <time-window-size value="0.001" />
     <max-time value="10"/>
     <participants first="Fluid" second="Calculix"/>
     <exchange data="Forces0" mesh="Calculix_Mesh" from="Fluid" to="Calculix"/>
     <exchange data="Displacements0" mesh="Calculix_Mesh" from="Calculix" to="Fluid" initialize="0" />

<max-iterations value="50"/>
<relative-convergence-measure limit="1e-4" data="Displacements0" mesh="Calculix_Mesh"/>
<relative-convergence-measure limit="1e-4" data="Forces0" mesh="Calculix_Mesh"/>
<extrapolation-order value="2"/>

 <acceleration:IQN-ILS>
   <data name="Displacements0" mesh="Calculix_Mesh"/>
   <preconditioner type="residual-sum"/>
   <filter type="QR1" limit="1e-6"/>
   <initial-relaxation value="0.1"/>
   <max-used-iterations value="50"/>
   <time-windows-reused value="10"/>
 </acceleration:IQN-ILS>
```

- <u>规定分区名称</u>
- 串行**隐式求解**
- 规定**时间步长**和**总时长**
- 规定交换信息形式**流体给固体力**、固体给流体**位移**
- 规定每组**最大迭代次数**、**收敛上限**

3、案例介绍 交界面时空耦合形式

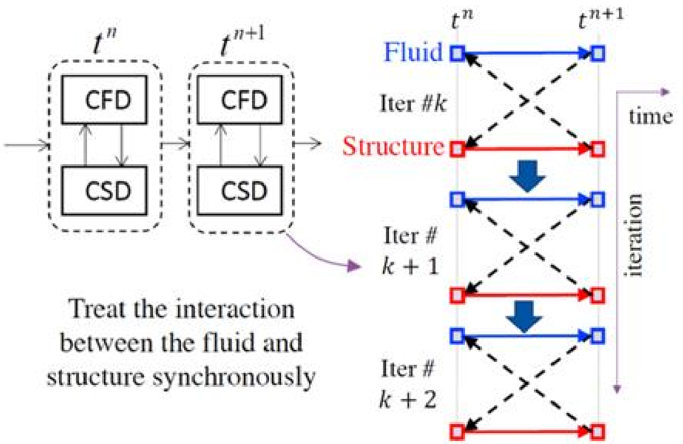
空间的耦合形式

使用随体网格ALE的方法



时间的耦合形式

Strongly/Implicit coupling scheme

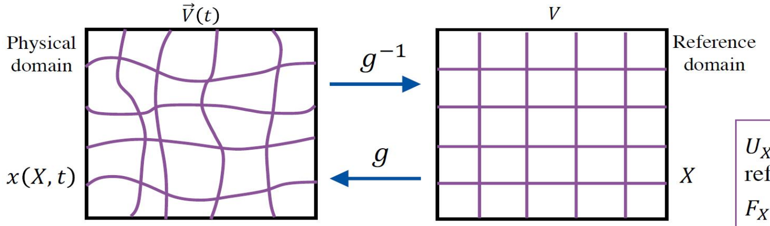


✓ Maintain conservation (more accurate and stable)
✓ More robust in handling large deformation

3、案例介绍 交界面时空耦合形式

空间的耦合形式

使用随体网格ALE的方法



$$\frac{\partial \vec{U}}{\partial t} + \nabla \cdot \vec{F}(\vec{U}) = 0$$

$$\vec{F} = c\vec{U}$$

$$\left.\frac{\partial \vec{U}_X}{\partial t}\right|_X + \nabla_X \cdot \vec{F}_X(\vec{U}_X) = 0$$

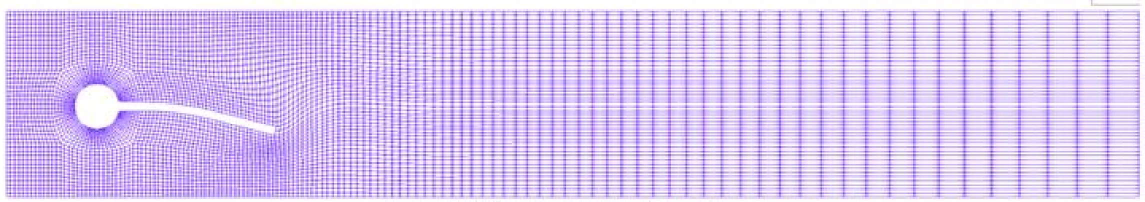
$$\vec{F}_X = gG^{-1}\vec{F} + \vec{U}_X G^{-1}\vec{V}_G$$

U_X: Variables in reference domain

F_X: Flux term

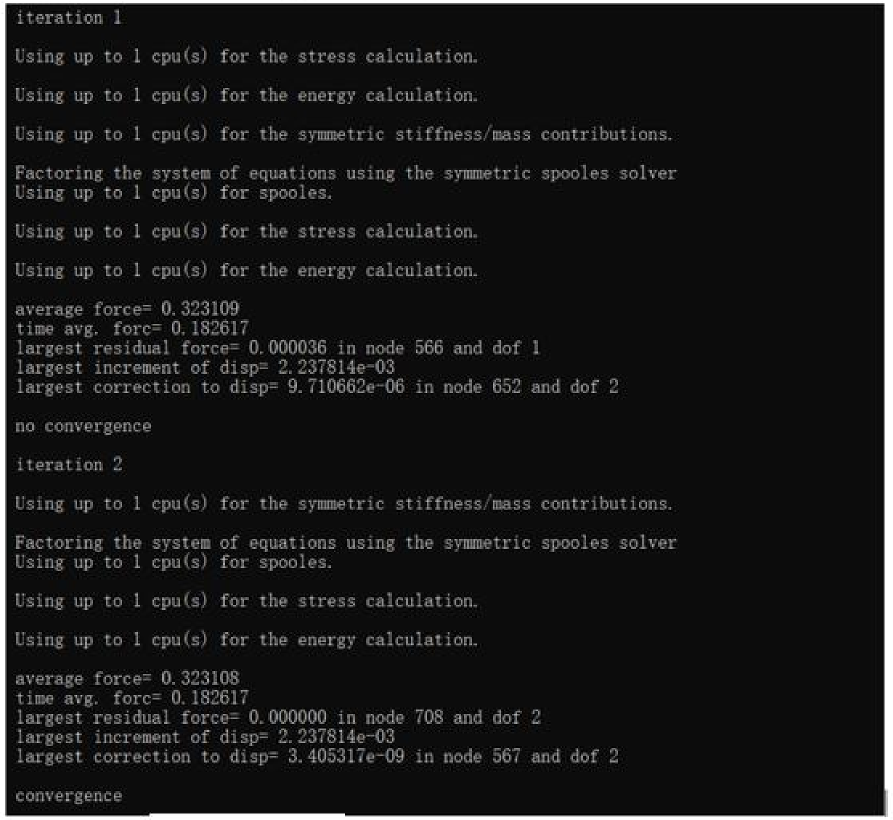
G^{-1}: Jacobi of $g(t)$

V_G: Mesh velocity



某个时刻的网格空间耦合示意图

时间的耦合形式

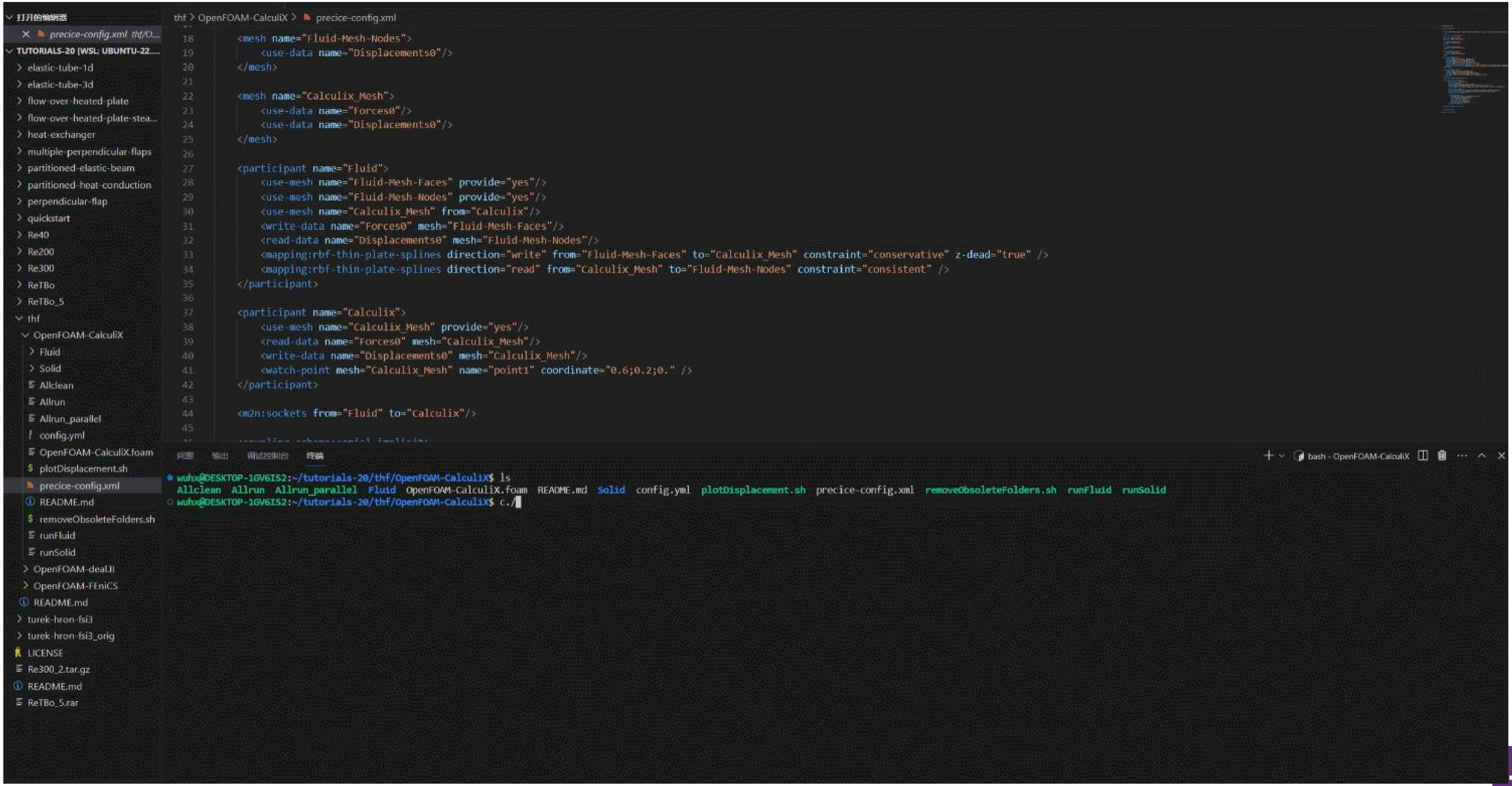


```
iteration 1

Using up to 1 cpu(s) for the stress calculation.

Using up to 1 cpu(s) for the energy calculation.

Using up to 1 cpu(s) for the symmetric stiffness/mass contributions.

Factoring the system of equations using the symmetric spooles solver
Using up to 1 cpu(s) for spooles.

Using up to 1 cpu(s) for the stress calculation.

Using up to 1 cpu(s) for the energy calculation.

average force= 0.323109
time avg. forc= 0.182617
largest residual force= 0.000036 in node 566 and dof 1
largest increment of disp= 2.237814e-03
largest correction to disp= 9.710662e-06 in node 652 and dof 2

no convergence

iteration 2

Using up to 1 cpu(s) for the symmetric stiffness/mass contributions.

Factoring the system of equations using the symmetric spooles solver
Using up to 1 cpu(s) for spooles.

Using up to 1 cpu(s) for the stress calculation.

Using up to 1 cpu(s) for the energy calculation.

average force= 0.323108
time avg. forc= 0.182617
largest residual force= 0.000000 in node 708 and dof 2
largest increment of disp= 2.237814e-03
largest correction to disp= 3.405317e-09 in node 567 and dof 2

convergence
```

某一时刻循环迭代示意图

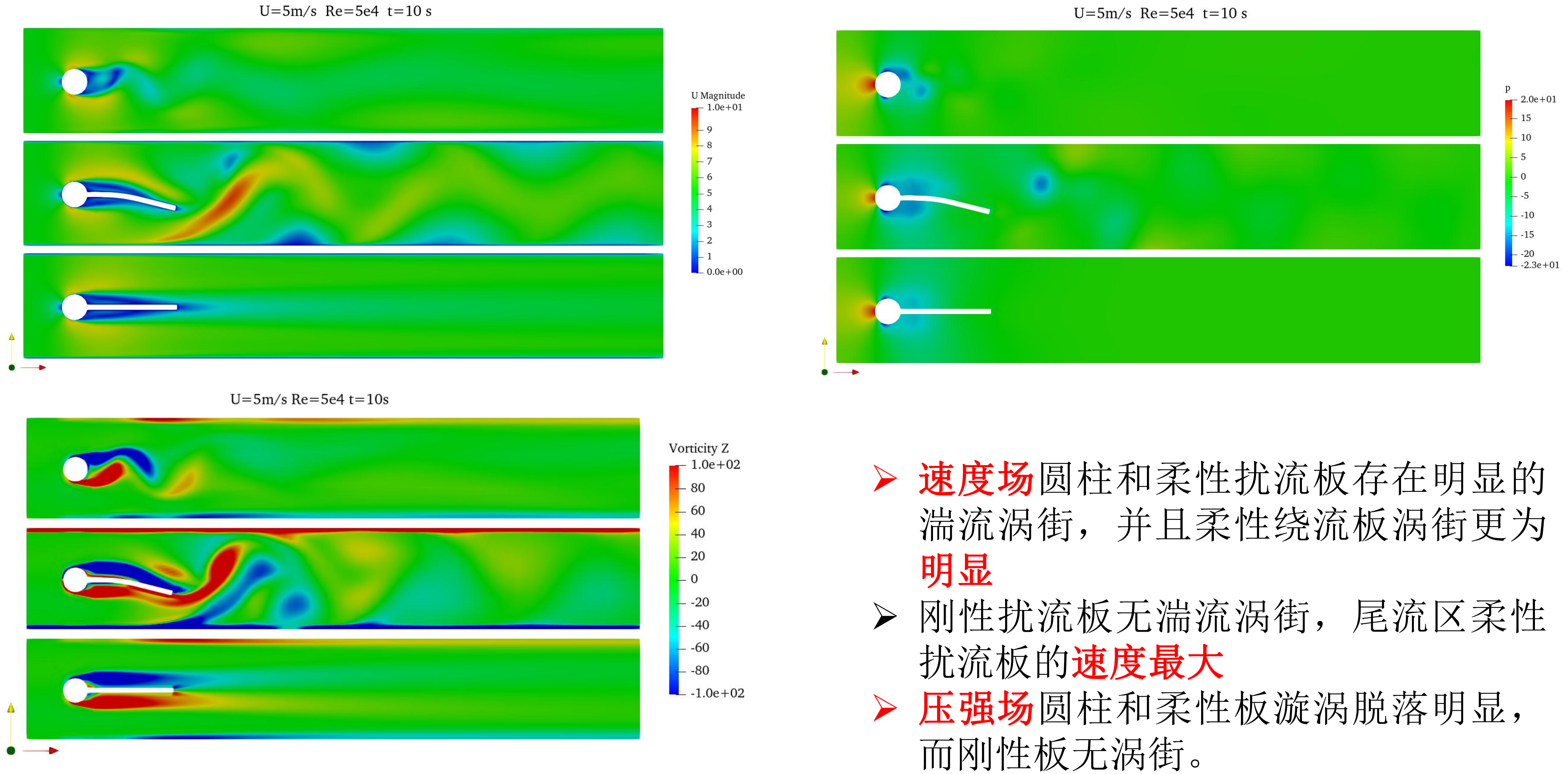
3、案例介绍

计算过程演示



速度、压强和涡量云图

为对比绕流板的效果，添加刚性绕流板和圆柱绕流进行比较



Re=5e4时速度场、压强场和涡量云图

- **速度场**圆柱和柔性扰流板存在明显的湍流涡街，并且柔性绕流板涡街更为**明显**
- 刚性扰流板无湍流涡街，尾流区柔性扰流板的**速度最大**
- **压强场**圆柱和柔性板漩涡脱落明显，而刚性板无涡街。

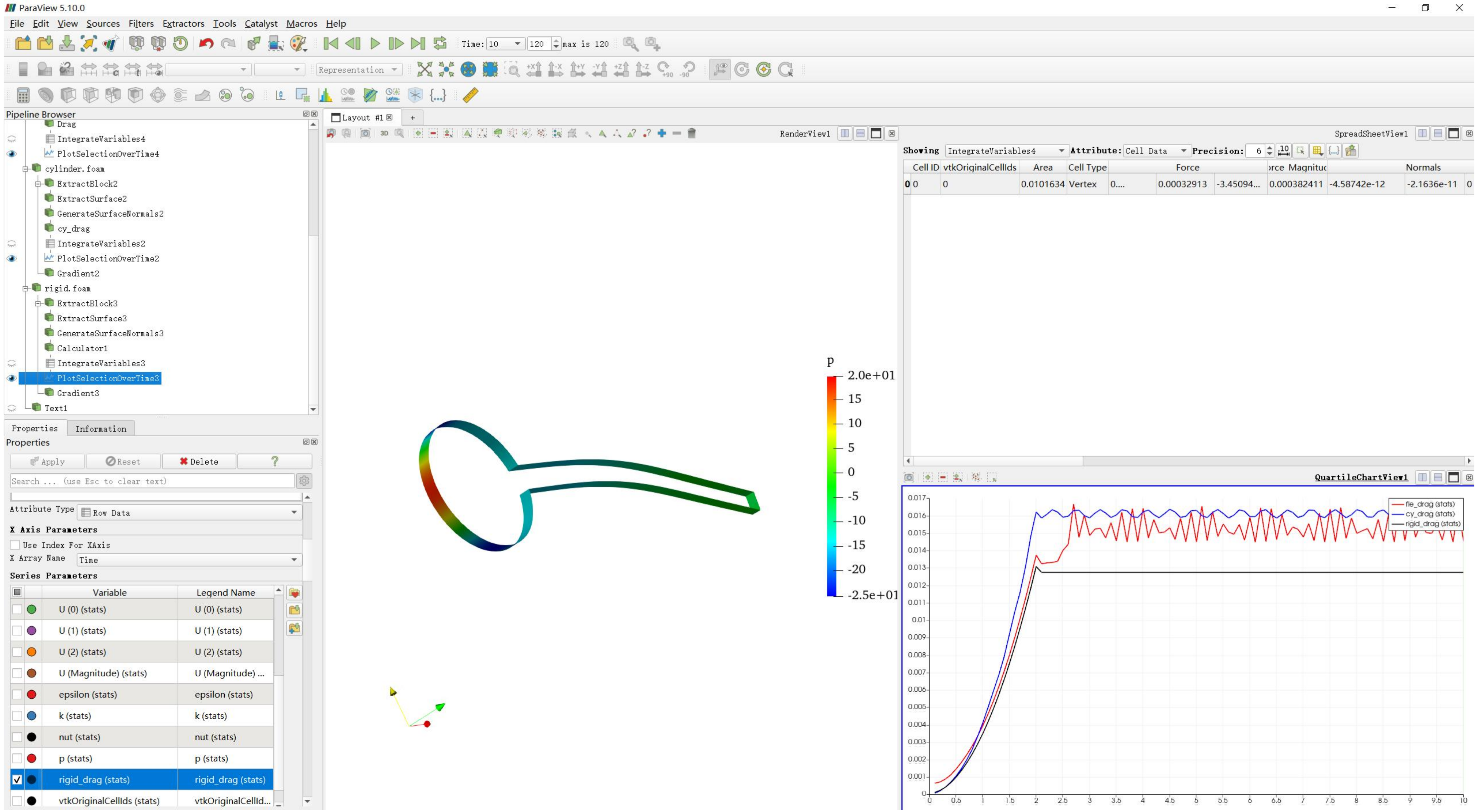
4 仿真数据可视化 速度、压强和涡量演化图

Re=5e4时速度场、压强场和涡量演化过程

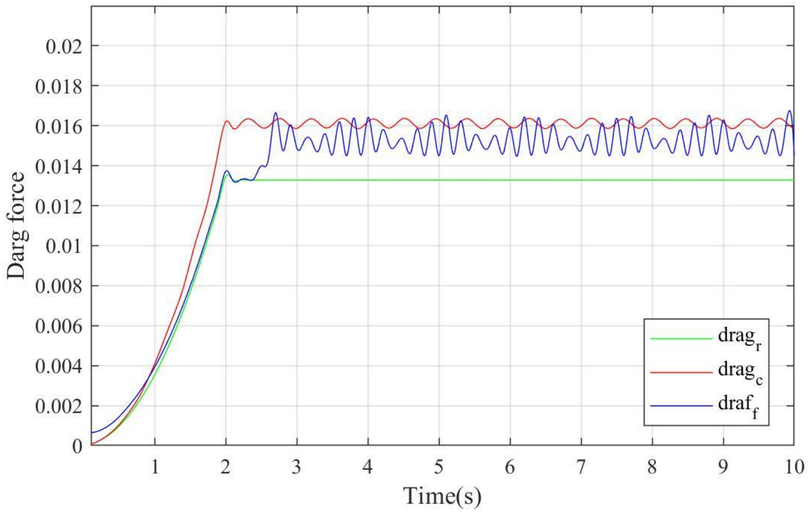
- **速度场**圆柱和柔性扰流板存在明显的湍流涡街，并且柔性绕流板涡街更为**明显**
- 刚性扰流板无湍流涡街，尾流区柔性扰流板的**速度最大**
- **压强场**圆柱和柔性板漩涡脱落明显，而刚性板无涡街。

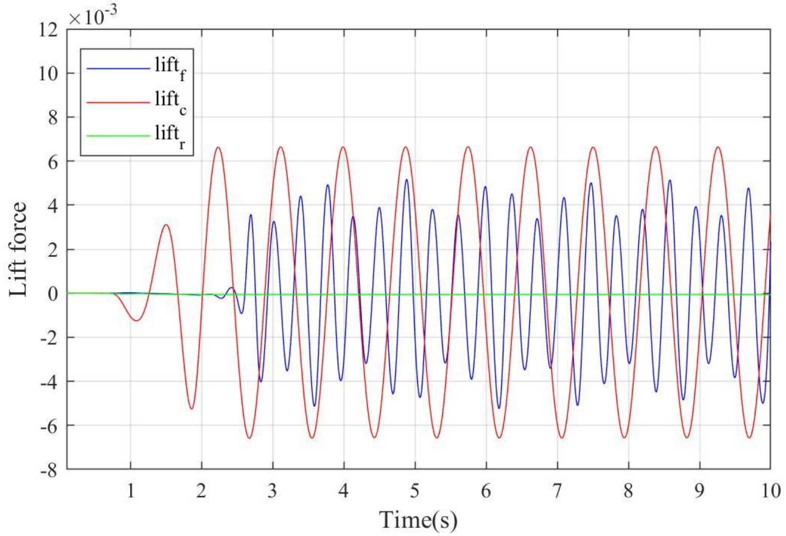
4 仿真数据可视化 提取升阻力

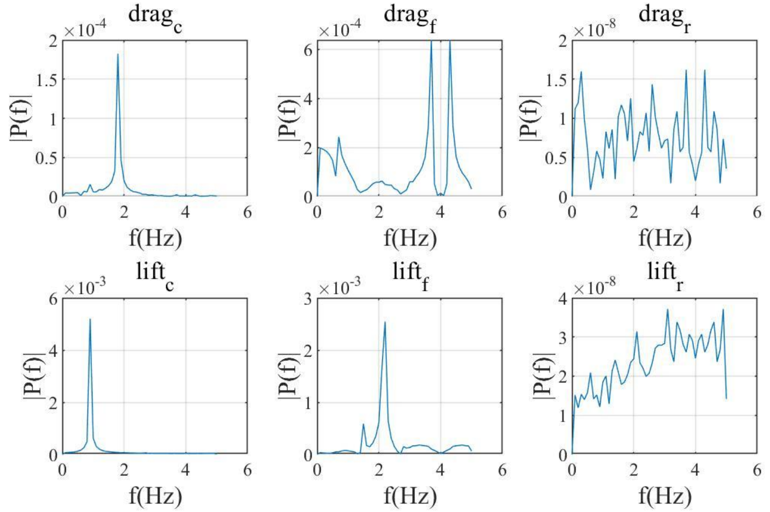
通过Paraview提取圆柱和绕流板表面的升阻力



4 仿真数据可视化 升阻力对比







- 三种结构的升阻力对比图，红色为圆柱、蓝色为柔性绕流板和绿色为刚性绕流板
- **刚性和柔性板**可以显著**减小**圆柱的升、阻力
- 刚性板阻力系数曲线**无波动**部分，这与观察到刚性板**无漩涡脱落**的现象吻合。
- 刚性板升力系数均值接近于0，说明刚性板**压强场**在升力向**分布接近对称**，无涡脱生成。
- 柔性板升阻力曲线的**振荡频率**比圆柱的**快**，且**频率成分**更为**丰富**。

5 结论与后续

结论

- 扰流板能**显著**改变高雷诺数下圆柱绕流的**压强场、速度场和涡量场**
- 柔性板能**提高**涡脱频率和**丰富**脱落的频率成分
- 刚性和柔性**扰流板**都能**减少**圆柱的升、阻力

后续

- 后续对柔性板进行**优化**、实现更好的**减阻效果**
- 探究其他范围雷诺数下**扰流板**对**圆柱涡脱特征**的影响
- 探究扰流板对**柔性圆柱**的涡脱特征的影响

清華大學深圳国际研究生院
Tsinghua Shenzhen International Graduate School

THANK YOU

汇报人：吴华晓